Aplication of Flotran CFD in ANSYS

Ing. arch. Ing. Milan Palko, PhD.

Slovak University of Technology in Bratislava, Faculty of Civil Engineering
Radlinského 11, 813 68 Bratislava, Slovak Republic
milan.palko@stuba.sk

Abstract

Simulation program ANSYS. The aerodynamic simulation with CFD (Computational Fluid Dynamics). Finite element method (FEM). Volume element method (VEM). The fluid flow problem is defined by the laws of conservation of mass, momentum, and energy. Eight turbulence models in FLOTRAN CFD. The geometric parameters of fluid dynamic simulation are described by finite network using many points. In ANSYS system there are two basic elements for FLORTAN - CFD. For tasks solved in plane FLUID 141 is used and for tasks solved in space FLUID 142 is applied. Simplified modelling of net for insect with support of real constant.

Keywords

CFD, ANSYS, double-skin transparent facade.

Introduction

There is a new way based on computer simulation. In my case, the simulation programme ANSYS based upon variation method of finite elements supported by FLOTRAN CFD module has been used for this method.

For mathematical description of fluid flow (air, liquid), mathematic variation methods are used represented by the two most significant ones:

- Finite Element Method - FEM,
- Volume Element Method - VEM.

The finite volume method is the most used airflow method. Presently, the finite element method reaches the level of the finite volume method due to progress and modification of mathematic solving elements. The choice of mathematic method (or the program using the method) was mainly influenced by the fact that SLOVAK UNIVERSITY OF TECHNOLOGY had purchased the licence of ANSYS programme, which enables this kind of simulation.The fluid flow problem is defined by three laws [1]:

- conservation of mass,
- conservation of momentum,
- conservation of energy.

Law of conservation of mass - continuity equation

From the law of conservation of mass law comes the continuity equation:

$$\frac{\partial \rho}{\partial t}+\frac{\partial\left(\rho V_x\right)}{\partial x}+\frac{\partial\left(\rho V_y\right)}{\partial y}+\frac{\partial\left(\rho V_z\right)}{\partial z}=0 \tag{1}$$

where

V_x, V_y, V_z components of the velocity vector in the x, y and z directions [m/s],

ρ density [kg/m^3],

t time [s],

x, y, z global Cartesian coordinates [m].

If the compressible algorithm is used, an ideal gas is assumed:

$$\rho=\frac{P}{RT} \tag{2}$$

thereout resulting:

$$\frac{\partial \rho}{\partial P}=\frac{1}{RT} \tag{3}$$

where

R gas constant [-],

T temperature [K].

Momentum equation

The momentum equations, without further assumptions regarding the properties, are as follows (Navier-Stokes equation) [1]:

$$\begin{aligned}\frac{\partial \rho V_x}{\partial t}+\frac{\partial\left(\rho V_x V_x\right)}{\partial x}+\frac{\partial\left(\rho V_y V_x\right)}{\partial y}+\frac{\partial\left(\rho V_z V_x\right)}{\partial z}=\rho g_x-\frac{\partial P}{\partial x}+\\+R_x+\frac{\partial}{\partial x}\left(\mu_e\frac{\partial V_x}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_e\frac{\partial V_x}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu_e\frac{\partial V_x}{\partial z}\right)+T_x\end{aligned} \tag{4}$$

$$\begin{aligned}\frac{\partial \rho V_y}{\partial t}+\frac{\partial\left(\rho V_x V_y\right)}{\partial x}+\frac{\partial\left(\rho V_y V_y\right)}{\partial y}+\frac{\partial\left(\rho V_z V_y\right)}{\partial z}=\rho g_y-\frac{\partial P}{\partial y}+\\+R_y+\frac{\partial}{\partial x}\left(\mu_e\frac{\partial V_y}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_e\frac{\partial V_y}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu_e\frac{\partial V_y}{\partial z}\right)+T_y\end{aligned} \tag{5}$$

$$\begin{aligned}\frac{\partial \rho V_z}{\partial t}+\frac{\partial\left(\rho V_x V_z\right)}{\partial x}+\frac{\partial\left(\rho V_y V_z\right)}{\partial y}+\frac{\partial\left(\rho V_z V_z\right)}{\partial z}=\rho g_z-\frac{\partial P}{\partial z}+\\+R_z+\frac{\partial}{\partial x}\left(\mu_e\frac{\partial V_z}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_e\frac{\partial V_z}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu_e\frac{\partial V_z}{\partial z}\right)+T_z\end{aligned} \tag{6}$$

where

g_x, g_y, g_z components of acceleration due to gravity [m/s^2],

ρ density [kg/m^3],

μ_e effective viscosity [N.s/m^2],

R_x, R_y, R_z distributed resistances [kg.m^2/.s^2],

T_x, T_y, T_z viscous loss terms [kg.m^2/.s^2].

For a laminar case, the effective viscosity μ_e is merely the dynamic viscosity.

Energy equation

The conservation of energy can be expressed in terms of the stagnation (total) temperature, often useful in highly compressible flows, or the static temperature, appropriate for low speed incompressible analyses. The complete energy equation is solved in the compressible case with heat transfer. In terms of the total (or stagnation) temperature, the energy equation is [1]:

$$\begin{aligned}\frac{\partial}{\partial t}\left(\rho C_p T_0\right)+\frac{\partial}{\partial x}\left(\rho V_x C_p T_0\right)+\frac{\partial}{\partial y}\left(\rho V_y C_p T_0\right)+\\+\frac{\partial}{\partial z}\left(\rho V_z C_p T_0\right)=\frac{\partial}{\partial x}\left(K\frac{\partial T_0}{\partial x}\right)+\frac{\partial}{\partial y}\left(K\frac{\partial T_0}{\partial y}\right)+\\+\frac{\partial}{\partial z}\left(K\frac{\partial T_0}{\partial z}\right)+W^v+E^k+Q_v+\Phi+\frac{\partial P}{\partial t}\end{aligned} \tag{7}$$

where

C_p specific heat [J/(kg.K)],

T_0 total temperature [K],

K thermal conductivity [W/(m.K)],

W^v viscous work term [J],

Q_v volumetric heat source [J],

Φ viscous heat generation term [J],

E^k kinetic energy [J].

The energy equation for the incompressible case may be derived from the one for the compressible case by neglecting the viscous work (W^v), the pressure work, viscous dissipation (f), and the kinetic energy (E^k). As the kinetic energy is neglected, the static temperature (T) and the total temperature (T_0) are the same. The energy equation now takes the form of a thermal transport equation for the static temperature [1]:

$$\frac{\partial}{\partial t}\left(\rho C_p T\right)+\frac{\partial}{\partial x}\left(\rho V_x C_p T\right)+\frac{\partial}{\partial y}\left(\rho V_y C_p T\right)+ \\ +\frac{\partial}{\partial z}\left(\rho V_z C_p T\right)=\frac{\partial}{\partial x}\left(K\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(K\frac{\partial T}{\partial y}\right)+ \\ +\frac{\partial}{\partial z}\left(K\frac{\partial T}{\partial z}\right)+Q_v \tag{8}$$

The momentum equations, without further assumptions.

Turbulence

There are eight turbulence models available in FLOTRAN. The model acronyms and names are as follows:

- Standard *k-ε* Model,
- Zero Equation Model,
- RNG - (Re-normalized Group Model),
- NKE - (New *k-ε* Model due to Shih),
- GIR - (Model due to Girimaji),
- SZL - (Shi, Zhu, Lumley Model),
- Standard *k-ω* Model,
- SST - (Shear Stress Transport Model).

Geometric parameters

The geometric parameters of fluid dynamic simulation are described by finite network using many points. In ANSYS system there are two basic elements for FLORTAN - CFD [2]. For tasks solved in plane FLUID 141 is used and for tasks solved in space FLUID 142 is applied. On such elements it is possible to solve the speed of fluid flow, pressure distribution and thermal effects. The element FLUID 141 (Fig. 1) can be used for modelling of steady or non-steady fluid/thermal systems. Using the element in FLORTAN - CFD the calculation for fluctuant and thermal distribution through area of elements forming the network is carried out. FLUID 141 also can be used at interaction of fluid and mass. On element the speeds gained on basis of the law of momentum preservation, the pressures rising from the law of mass conservation, the temperature from the law of energy conservation are calculated. The unknown values create the levels of element disposal, which are defined by global iteration. A FLUID 141 element has six degree of freedom in each point. Degree of freedom are [2]:

- velocity in direction (VX, VY),
- press (PRES),
- temperature (TEMP),
- turbulent kinetic energy (ENKE),
- turbulent kinetic energy dissipation rate (ENDS).

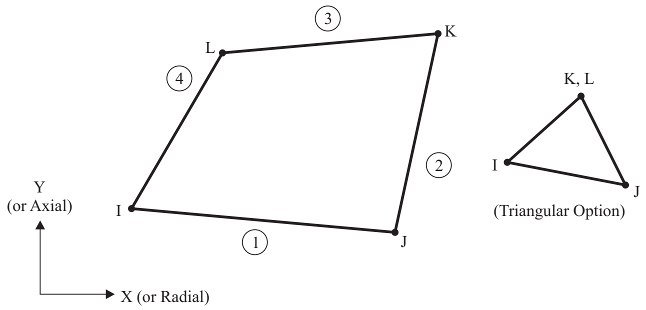
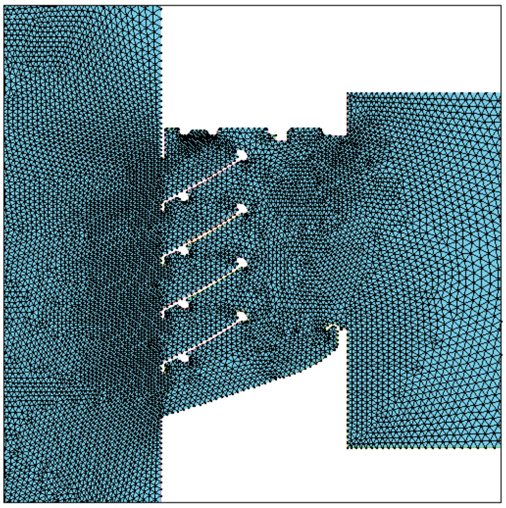


Fig. 1 Element FLUID 141



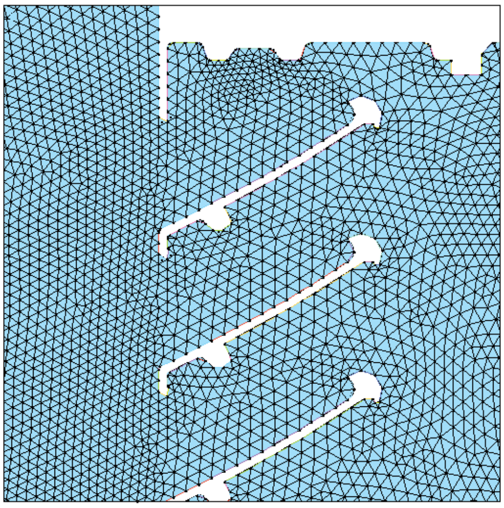


Fig. 2 Mashing with element FLUID 141 (input channel of double skin façade)

The element is defined by three nodes (triangle) or four nodes (quadrilateral) and by isotropic material properties. The coordinate system is selected according to the value of KEYOPT, and may be either Cartesian, axisymmetric, or polar.

Approximation of net model

Due to too much detail it was necessary to simplify the net model. If we wanted to model the net against insects based on geometry (while the wire thickness is 0,5 mm) it would be very demanding relating to detail of network compression (considering the model size). From the above given reasons I used the approximation of model following the real material constants, which are provided by ANSYS programme. I used the constant ,,K,, (Head loos) for my investigation. However, I had to determine the material constant by inverse way, as it had not existed for the given network. The aerodynamic coefficients of local resistance are determined in a large geometry spectrum based on experimental measurements. Following the known value of network resistance I was able to determine ,,K,, constant. The simulation model was as follows: The part, for which ,,K,, constant was determined empirically, was placed into the pipe (Fig. 3). After the simulation was carried out, its aerodynamic resistance was determined following the total and aerodynamic pressures. ,,K,, value was being changed until the resistance from the simulation and the experiment were identical. The gained ,,K,, coefficient was then used the material constant into model of input channel. [3]

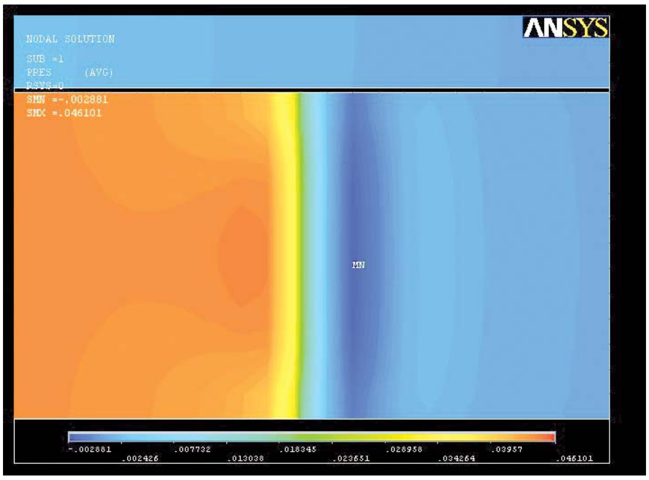


Fig. 3 Model of net in tube - pressure fields

Conclusions

Use FLOTRAN-CFD can solve a variety of applications in building construction. Fig. 3 presents the approximate model mosquito nets. Fig. 4 shows the velocity fields in the input channel of double skin façade.

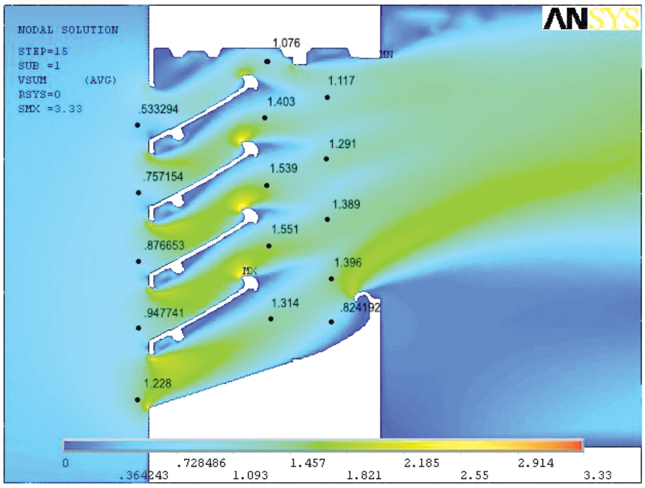


Fig. 4 Velocity field of inlet channel for input velocity 0,4 m/s

References

[1] Crochet, M.J.; Davies, A.R.; Walters, K.: *Numerical Simulation of Non-Newtonian Flow*, Elsevier Science Publishers B.V. (1984).

[2] Gartling, D.K.: *"Finite Element Methods for Non-Newtonian Flows"*, report SAND92-0886, CFD Dept., Sandia National Laboratories, Albuquerque, NM (1992).

[3] White, F.M.: *Viscous Fluid Flow*, Second Edition, McGraw-Hill, New York (1991).